Analysis of Flowfields Over Four-Engine DC–X Rockets

T.S. Wang/ED32
205–544–0503
E-mail: Ten-see.Wang@msfc.nasa.gov

The objective of this study is to validate a computational fluid dynamics (CFD) methodology for the aerodynamic performance of an advanced conical launch vehicle configuration, the Delta Clipper-Experiment (DC–X) rocket, with emphasis on multiple-engine plume-on effects. Both wind tunnel and flight test data are used for validation. Computational base-drag characterization in the critical subsonic regime is the main goal since until recently, almost no multiple-engine data exists for a conical launch vehicle configuration.

For launch vehicles using clustered engines, it is well known that the base environment significantly affects the overall drag and integrity of these vehicles. Hence, it becomes very important to be able to predict the base drag during the vehicle design phase. While empirical equations, wind tunnel and historical flight test data are still an integral part of the design process, CFD-based methods have emerged as a new tool. The DC–X tests are unique in that the flight vehicle and the cold-flow model have satisfied the basics of the scaling law and can therefore complement each other's limitations in terms of the measurement. A systematic validation process of both tests is an opportunity to further prepare CFD as a powerful design tool to support future X–33 reusable launch vehicle aerodynamic performance characterization and vehicle design refinement and optimization.

A three-dimensional, viscous flow, CFD design code FDNS is used for this purpose. While previous benchmarks[1] covered a range of nozzle exit-to-ambient-pressure-ratio from 5 to 510, this work covers the critical lower spectrum of pressure-ratio from 1.2 to 1.7. Parametric studies using high-order difference schemes are performed for the cold-flow tests while finite-rate afterburning chemical kinetics are used for the flight tests. The computations are performed on MSFC's CRAY-YMP and CRAY-Trident machines.

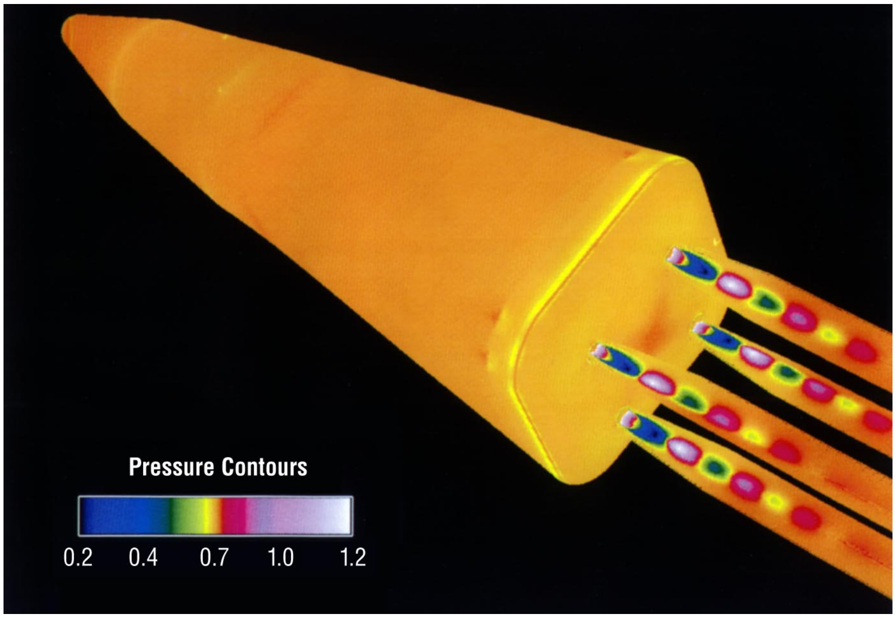

FIGURE 59.—The computed DC–X ascent flight test pressure contours.

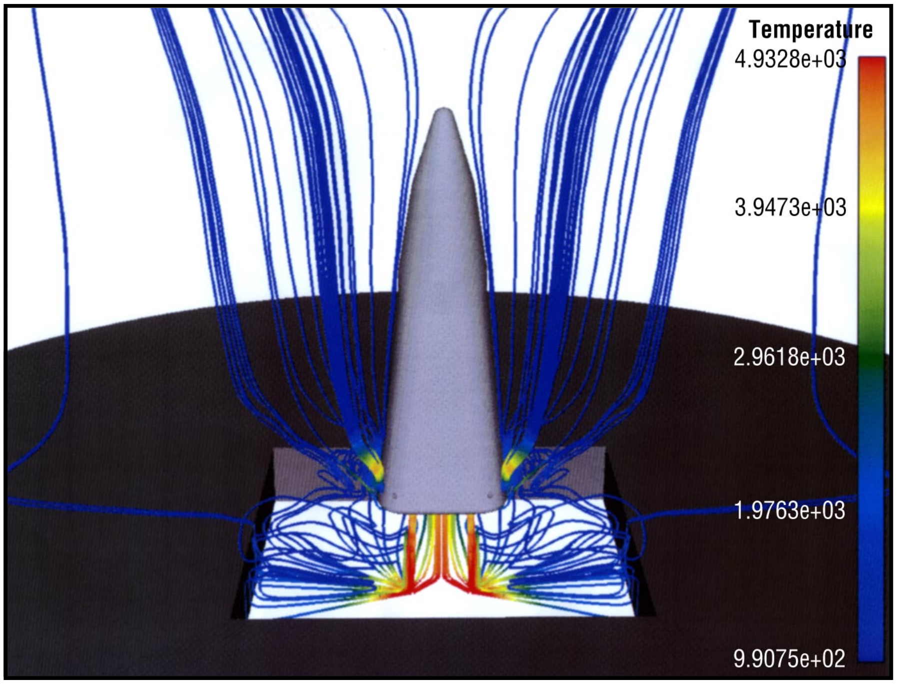

FIGURE 60.—The computed particle traces (colored by temperature) for DC–X landing on grate.

The computed DC–X ascent vehicle total drag, forbody and aftbody and base-surface pressure coefficients compared favorably with those of the test data. The result demonstrates that with adequate grid density distribution, high-order difference scheme, finite-rate afterburning kinetics to model the plume chemistry, and an appropriate turbulence model to describe separated flows, plume/air mixing, and boundary layers, the CFD methodology can be used to predict low-speed aerodynamic performance for reusable rocket designs. In addition, FDNS is also benchmarked for the DC–X in-ground effects. Reasonable base heat flux and aerodynamic force are predicted and compared to the flight tests. This validated computational methodology is later used to study the flow anomaly of DC–XA landing on grate.

[1]Wang, T.S.: “Grid-Resolved Analysis of Base Flowfield for Four-Engine Clustered Nozzle Configuration.” *Journal of Spacecraft and Rockets,* vol. 33, no. 1, Jan.–Feb. 1996, pp. 22–29.

Sponsor: McDonnell Douglas—NASA Marshall Space Flight Center Cooperative Agreement for Reusable Launch Vehicle/ X–33 Technology Development Program

Biographical Sketch: Dr. Ten-See Wang is currently a team leader of the Computational Analysis Team in the Fluid Dynamics Branch. He received his Ph.D. degree from Louisiana State University in 1980. He had previously been affiliated with SAIC, Continuum, SRA, and SECA, Inc.. His recent work has been applying CFD methods for propulsion system and launch vehicle environment.